

Numerical simulation and experimental validation of the hydrodynamics in a 350 kW bubbling fluidized bed combustor

Embarek Belhadj¹ · Stephen Chilton² · William Nimmo² · Hubert Roth¹ · Mohamed Pourkashanian²

Received: 6 August 2015 / Accepted: 23 December 2015 / Published online: 28 January 2016
© The Author(s) 2016. This article is published with open access at Springerlink.com

Abstract This paper presents experimentally validated three-dimensional numerical simulation of a 350 kW pilot-scale bubbling fluidized bed combustor, which has been developed by using commercial CFD software package, Fluent 14.5. The solid particle distribution has been simulated by using the multiphase Euler–Euler Approach. The gas–solid momentum exchange coefficients were calculated by using Syamlal and O’Brien drag functions. The CFD model is created as the realistic representation of the actual pilot-scale bubbling fluidized bed. All simulations are performed in transient mode for an operation time of about 350 s. The experimental study is performed with silica sand particles with mean particle size of 0.6 mm and density of 1639 kg/m³. The bed was filled with particles up to a height of 0.30 m. The same conditions are used for the simulations. The present work combines both experimental and computational studies, where the CFD-Simulation results are compared to those obtained by experiments. The predicted simulation results of minimum fluidization velocity and pressure drop values of the pilot-scale bubbling fluidized bed combustor have good agreement with the experimental measurements.

Keywords Fluidized bed · Sewage sludge combustion · Hydrodynamics · Numerical simulation · Two-fluid model

Introduction

Fluidized beds are used in a wide range of industrial applications, covering many sectors including chemical, combustion and energy industries. This variety of applications by fluidized bed systems has the importance of this technology enormously increased. Due to uniform particle mixing and large areas of contact between different phases generated by intensive mixing, fluidized beds have become an important asset in the field of combustion. In the fluidized bed combustor, the fuel particles are suspended and burnt with an intensive mass and heat exchange of hot sand particles and combustion air as well as mass transfer and reactions between gas and fuel particles. In this process, the resulting combustion heat is directly absorbed by the sand bed. This thermal energy stored in the sand particles leads to a homogeneous temperature distribution throughout the fluidized bed and prevents the formation of temperature peaks in bed surface areas [1]. Therefore the heat transfer efficiency in the fluidized bed combustors is strongly dependent on the fluidization quality, and thus is necessary to understand the most important hydrodynamic parameters of the mixtures such as minimum fluidization velocity and maximum bed pressure drop in order to ensure an optimal operating conditions and a complete combustion with minimal pollutant formation in combustion processes.

Bubbling fluidized bed combustor offers a number of advantages compared to other traditional technologies including better heat transfer characteristics and lower temperature requirements. This results in lower nitrogen oxide (NO_x) formation, which can be further lowered by the introduction of moderated secondary over fire air. Another important advantage of fluidized beds is their ability to incinerate a wide variety of materials. A fluidized bed relies on residual heat being retained within the bed

✉ Embarek Belhadj
EBelhadj@gmx.de

¹ Department of Electrical Engineering and Computer Science, Institute of Automatic Control Engineering, University of Siegen, Siegen, Germany

² Energy Technology and Innovation Initiative, University of Leeds, Leeds LS2 9JT, UK

and surrounding lining of the combustion area. Therefore when fuels of varying particle size, moisture content, ashing potential and calorific value are introduced, a fluidized bed can completely combust the material whilst utilising the fuels energy to its maximum potential. All these points culminate to a technology which is desired when handling less than desirable fuels [1–3].

During the last few years, fluidized bed combustion has been increasingly utilised in the field of combustion of Biomass, sewage sludge and low grade brown coals in order to decrease the emissions and minimising the environmental impact in energy production. Research, development and design of fluidized bed reactors has been focused towards achieving a better understanding of the behaviour of the bed material during the combustion process. But the complex physical and chemical process inside the fluidized bed is still not well understood [1, 2]. Because of the multivariable and complex nonlinear behaviour of the fluidized beds and the many solid particle interactions, the modelling of fluidized bed reactors to simulate the hydrodynamics of gas–solid particles is very challenging. Computational Fluid Dynamics (CFD) is the most common numerical technique to simulate multiphase flow. CFD has been developed especially for the simulation of the flow behaviour, and has proven its use in the investigation and optimization of many processes. These numerical methods have the advantage, where the experiments are not possible to be undertaken in a real system, because of the high costs, time required and complicity. The objective of CFD-Simulation is to identify complex flow problems in the construction as well as in existing systems and to help optimising the processes. The use of CFD program packages in design and analysis of industrial flow processes has significantly increased in the last decade, especially in the field of combustion and energy industries. There have been numerous studies carried out and considerable progress made in the last few years in the area of hydrodynamic modelling of gas–solid particle in the fluidized bed with the use of CFD simulation software [4–6].

CFD as a method of analysis is becoming an important tool to advance our understanding of the hydrodynamics in fluidized beds. Nevertheless, CFD is still at the validation stages for modelling multiphase flow, and more improvements regarding the flow dynamics and computational models are required to make it a more reliable tool in designing of large scale industrial reactors [3].

In the literature, there are two different numerical approaches for modelling the hydrodynamics of gas–solid two-phase flow with CFD simulation: the Euler–Euler and the Euler–Lagrange method [7]. The Euler–Euler method treats each phase as an interpenetrating continuum. In the Euler–Lagrange approach, the gas phase is treated as continua while the solid phases are treated as discrete

particles. There are also many drag models that have been developed to investigate the interaction between gas and solid particles in fluidized bed, such as the Syamlal and O'Brien, Wen and Yu and Gidaspow drag models [1, 4, 7]. Unfortunately, in only a limited number of studies have researchers combine both numerical and experimental investigations on the hydrodynamics of a gas–solid fluidized beds.

Taghipour et al. [8] investigated the hydrodynamics of a two-dimensional gas–solid fluidized bed reactor using a Syamlal and O'Brien, Wen and Yu and Gidaspow drag models, and found that the predicted pressure drops with Syamlal and O'Brien and Gidaspow drag models are in good agreement with the experimental measurements at a higher superficial velocity than the minimum fluidization velocity.

Hamzehei [9] compared the CFD simulation predicted results using the Syamlal and O'Brien drag model to the experimentally measured pressure drop, and found that the model predictions were in good agreement with the experimental data.

Ramesh et al. and Almuttahir [10, 11] investigated the hydrodynamic fluidized bed results predicted from the Arastoopour, Gidaspow and Syamlal and O'Brien drag models, the Syamlal and O'Brien drag model was found to provide better predictions.

Esmaili and Mahinpey [12] have compared the results from the simulations with eleven different drag models with respect to minimum fluidization velocity, and found that Syamlal and O'Brien gives better prediction when compared with other models. In addition, the Syamlal and O'Brien drag is able to more accurately predict the minimum fluidization velocity. They also found that three-dimensional (3D) simulations provide better results than two-dimensional (2D) simulations compared with experiments.

Furthermore, one of the main difficulties to validate CFD models using experimental data is the computational effort and time needed to perform a detailed 3D simulations of the hydrodynamic behaviours in fluidized beds, especially for sizing from a pilot-scale fluidized bed reactors to large industrial units. Therefore, most of the CFD studies on hydrodynamics in fluidized bed reactors have been performed on a 2D small-scale laboratory rig.

The aim of the present study is therefore to fill this gap by developing a 3D numerical simulation of gas–solid flow in a pilot-scale bubbling fluidized bed reactor.

The numerical simulations of this bubbling fluidized bed were performed using the Eulerian–Eulerian CFD model incorporating the kinetic theory of granular flow in order to simulate the gas–solid flow behaviour. Here, results obtained through CFD simulations calculated by using the Syamlal and O'Brien drag functions are compared with the available experimental results from the test rig.



The main objective of these investigations carried out at pilot-scale bubbling fluidized bed is to assess the accuracy of the pressure drop and minimum fluidization velocity results predicted from the simulations compared to experimental measurements at different superficial gas velocities, which are known as the most important parameters that characterize the gas–solid fluidization quality, and finally to check if the Syamlal and O'Brien model predicts the fluidization conditions correctly compared to the measurements obtained from the pilot-scale bed.

Various superficial gas velocities, 0.5, 1.0, 1.25 and 1.5 m/s were examined to determine maximum pressure drop.

CFD modelling and simulation

In this study, the numerical simulations of gas–solid particles interactions in a three-dimensional fluidized bed reactor were carried out. The geometric dimensions for the model are similar to the pilot test rig. The modelling and meshing were developed by using SolidWorks and ICEM CFD software. The geometrical model is meshed using a structured hexahedral grid with approximately 165,000 cells (502,000 faces). The total number of the computational grid elements is 182,000.

The simulation geometry of the gas–solid fluidized bed is shown in Fig. 1. The multiphase model was implemented in the commercial CFD code FLUENT 14.5 using the Eulerian–Eulerian approach, thereby both phases, gas and solid are treated as interpenetrating continua but separately. The gas–solid momentum exchange coefficients were calculated by using Syamlal and O'Brien drag functions. The internal dimensions of the fluidized bed reactor are 0.42×0.38 m, which give a bed area of 0.16 m^2 , and the height, including freeboard, is 5.0 m. The initial bed height was 0.30 m. Sand particles with a mean size of 0.6 mm and density of 1639 kg/m^3 were used. These values are consistent with those obtained from the experimental rig. The same modelling parameters were used for all the cases with varying only the inlet gas velocity. The inlet superficial gas velocities are set to 0.5, 1.0, 1.25, and 1.5 m/s. The simulations were performed in transient mode for a time span of 350 s. The computational fluid dynamics (CFD) model have been developed to investigate how the inlet air velocity profile affects the fluidization behaviour of the sand particles in the pilot-scale bubbling fluidized bed combustor, particularly the pressure drop across the bed of this solid material and finally choose the correct parameters for a bubbling flow regime.

The simulation results of pressure drop and fluidization velocity predicted by the developed CFD model are

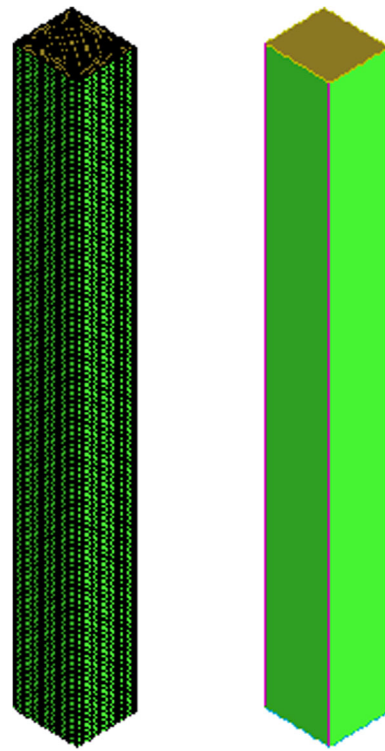


Fig. 1 The numerical mesh of the experimental BFBC

validated against experimental measurements obtained from the bubbling fluidized bed combustor.

The Eulerian–Eulerian model equations for the gas–solid flow

In the Eulerian–Eulerian multiphase model, both phases are treated as continuum. The governing equations solved for the current gas–solid system include the conservation of mass and momentum.

The continuity equation between gas and solid phases is give for each phase [6, 13].

The mass conservation of the gas phase (g) can be written as:

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g) + \nabla \cdot (\varepsilon_g \rho_g \mathbf{v}_g) = 0 \quad (1)$$

and the mass conservation of the solid phase (s) is:

$$\frac{\partial}{\partial t}(\varepsilon_s \rho_s) + \nabla \cdot (\varepsilon_s \rho_s \mathbf{v}_s) = 0 \quad (2)$$

where ρ_g and ρ_s are the density of gas and solid phases, and \mathbf{v}_g and \mathbf{v}_s are the velocity vectors for the gas and solid phases.

The ε_g and ε_s are the volume fractions of the gas and solid phases respectively which satisfy the relation.



$$\varepsilon_g + \varepsilon_s = 1 \quad (3)$$

The conservation equation for the momentum of gas phase is:

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g \mathbf{v}_g) + \nabla \cdot (\varepsilon_g \rho_g \mathbf{v}_g \mathbf{v}_g) = -\varepsilon_g \nabla p + \nabla \cdot \bar{\bar{\tau}}_g + \varepsilon_g \rho_g \mathbf{g} + K_{sg}(\mathbf{v}_s - \mathbf{v}_g) \quad (4)$$

and the conservation of momentum for the solid phase (s) is:

$$\frac{\partial}{\partial t}(\varepsilon_s \rho_s \mathbf{v}_s) + \nabla \cdot (\varepsilon_s \rho_s \mathbf{v}_s \mathbf{v}_s) = -\varepsilon_s \nabla p - \nabla p_s + \nabla \cdot \bar{\bar{\tau}}_s + \varepsilon_s \rho_s \mathbf{g} + K_{gs}(\mathbf{v}_g - \mathbf{v}_s) \quad (5)$$

The subscripts (g) and (s) stand for gas and solid phases, (ε) is the volume fraction, (ρ) is the density, (p) is the pressure shared by both phases gas and solid, (p_s) is the solid pressure, ($\bar{\bar{\tau}}_s$) is the stress tensor, (\mathbf{g}) is the gravity vector and ($K_{sg} = K_{gs}$) is the fluid–solid exchange coefficient.

Drag model

The interactions between solid particles and the continuous gas phase are described by a drag model.

Several drag models for the gas–solid interphase momentum exchange coefficient K_{gs} were reported in the literature [1].

The drag models which are more widely used are Syamlal and O'Brien, Gidaspow and Wen–Yu drag [11, 14, 15]. Syamlal and O'Brien drag function gives a better results compared to other drag models, and it is more suitable for predicting the hydrodynamics of gas–solid flow in fluidized beds [9, 11].

Therefore, Syamlal and O'Brien drag function has been applied in this study to describe the momentum exchange between phases.

This drag law is based on the measurements of the terminal velocities of particles in fluidized or settling beds, with correlations which are functions of the volume fraction and the relative Reynolds number [9, 15].

The gas–solid exchange coefficient has the form:

$$K_{gs} = \frac{3 \cdot \varepsilon_s \cdot \varepsilon_g \cdot \rho_g}{4 \cdot v_{r,s}^2 \cdot d_s} \cdot C_D \cdot \left(\frac{Re_s}{v_{r,s}} \right) \cdot |\mathbf{v}_g - \mathbf{v}_s| \quad (6)$$

where the drag function is given by:

$$C_D = \left(0.63 + \frac{4.8}{\sqrt{\frac{Re_s}{v_{r,s}}}} \right)^2 \quad (7)$$

and ($v_{r,s}$) is the terminal velocity correlation for the solid phase:

$$v_{r,s} = 0.5 \sqrt{(0.06 Re_s)^2 + 0.12 Re_s (2B - A) + A^2} + 0.5 \cdot A - 0.03 \cdot Re_s \quad (8)$$

with

$$A = \varepsilon_g^{4.14} \quad \text{and} \quad B = \begin{cases} 0.8 \cdot \varepsilon_g^{1.28} & \text{for } \varepsilon_g \leq 0.85 \\ \varepsilon_g^{2.65} & \text{for } \varepsilon_g > 0.85 \end{cases}$$

The relative solid Reynolds number of the solid phase is defined as:

$$Re_s = \frac{\rho_g \cdot d_s \cdot |\mathbf{v}_g - \mathbf{v}_s|}{\mu_g} \quad (9)$$

where (d_s) is the particle diameter and (μ_g) is dynamic viscosity of the gas.

Pressure drop and minimum fluidization velocity

In the field of combustion, the air flow rate plays a number of important roles; primarily providing a cushion of air which the bed will sit upon as well as mixing the bed material. This high mixing is the key factor in achieving a uniform combustion temperature as well as in the oxidation of combustion materials. In practice, the expansion of the bed materials in the combustor is almost controlled and limited by the pressure drop across the bed. Therefore, the hydrodynamic properties such as the bed pressure drop and minimum fluidization velocity are the most important parameters studied in the numerical modelling of fluidized beds. To achieve and maintain a stable fluidization, knowledge about the pressure drop across the bed and minimum fluidization velocity of the gas flow introduced from the bottom of the bed are required. Therefore the minimum fluidization air velocity to fluidize the sand particles and pressure drop are crucial hydrodynamic parameters for analysing the operation and design of fluidized bed combustors. The minimum fluidization velocity was determined experimentally by measuring the pressure drop through the bed of particles. The pressure drop is defined as the difference of absolute pressure under the bed to that of the area above the bed which is called the free-board [16, 17].

When the gas velocity is equal to the minimum fluidization velocity, the bed pressure drop becomes equal to its weight per unit volume and thus the pressure drop (Δp) across the bed in the fluidized condition is calculated as [18, 19]:

$$\Delta p = (1 - \varepsilon_{mf}) \cdot (\rho_s - \rho_g) \cdot g \cdot h_{mf} \quad (10)$$

where (g) is the gravity, (ε_{mf}) is the volume fraction occupied by the fluid at minimum fluidization velocity and (h_{mf}) is the initial height of the bed at this condition.



The transition of the rig from fixed to fluidized bed is controlled by the minimum fluidisation velocity (U_{mf}), which depends mainly on the fluidization material characteristics such as particle diameter and density, and is defined as [4, 20]:

$$U_{\text{mf}} = \frac{d_s^2 \cdot (\rho_s - \rho_g) \cdot g}{1650 \cdot \mu_g} \quad \text{for } Re_s \prec 20 \quad (11)$$

$$U_{\text{mf}}^2 = \frac{d_s \cdot (\rho_s - \rho_g) \cdot g}{24.5 \cdot \mu_g} \quad \text{for } Re_s \succ 1000 \quad (12)$$

The Reynolds number (Re_{mf}) at minimum fluidization velocity is given by the equation [21]:

$$Re_{mf} = \frac{\rho_g \cdot d_s \cdot U_{mf}}{\mu_g} \quad (13)$$

Fluidized bed experimental test rig

For the validation of the simulations, the experiments were performed on a pilot-scale bubbling fluidized bed combustor with a heat capacity of 350 kW.

This test facility incorporates a combustor section, an air cooled heat exchanger for cooling the flue gases, a cyclone and bag filter. This bubbling fluidized bed also includes a temperature and pressure measurement devices. The schematic of the pilot-scale test rig used in the current study is shown in Fig. 2.

All process parameters such as temperatures and pressures were recorded using thermocouples and pressure transmitters and stored every 5 s by a computer logger. The pressures were measured at five different locations, at the bottom (Plenum), below bed, above bed and in the free-board. The measured data obtained from the fluidized bed including volumetric flow rates of the fluidizing air and emission parameters were connected to a National Instrument module and then recorded and analyzed.

The combustor section has an overall dimensions of $1\text{ m} \times 1\text{ m} \times 5\text{ m}$ high and consists of a fluidized bed modules, a transition section, and an extended freeboard section. The mild steel casing is refractory lined throughout. The internal dimensions of the fluidized bed are $0.42 \times 0.38\text{ m}$. These dimensions enable minerals to be processed at rates up to 500 kg/h , and combustion of bio-fuels and wastes at up to 100 kg/h , depending on material type and plant operating conditions.

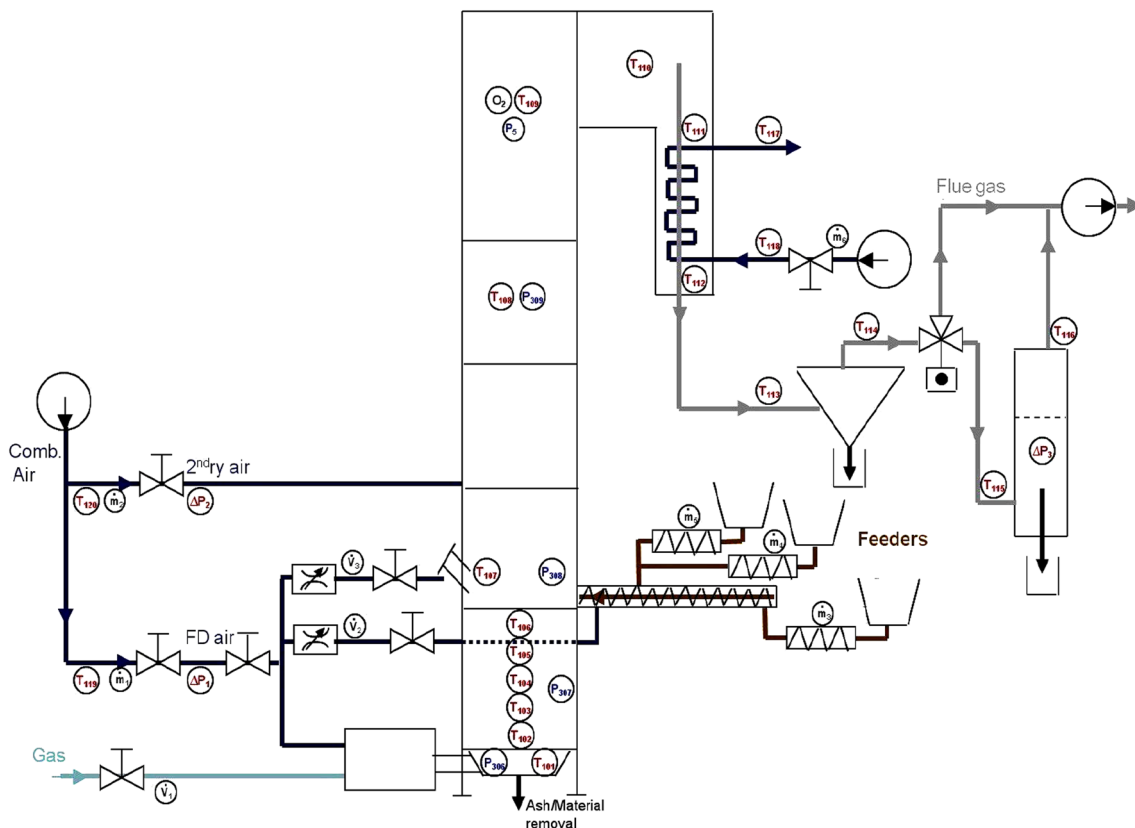


Fig. 2 Schematic of the used test bubbling fluidized bed including instrumentation

The plant can be operated with fluidized bed heights of up to 0.70 m, and fluidizing velocities of up to 3.5 m/s at typical combustion temperatures. The fluidized bed was filled with silica sand as bed material up to a height of 0.30 m. The particles had a density of 1639 kg/m^3 and an average mean diameter of 0.6 mm. The same conditions are used for the simulations.

This bubbling fluidized bed combustor operates in the bubbling fluidization regime. The fluidizing air was introduced from the bottom of the combustor. The bed material was fluidized by controlling the air flow rate.

The measured air mass flow rates were between 0.17 and 0.19 kg/s. The superficial inlet gas velocity through the bed can be calculated using $U_0 = Q/A$ and $\rho_g = m/v$, Q is the air flow rate in m^3/s and $A = 0.16 \text{ m}^2$ is the cross-sectional area of the bed.

Therefore, with the air density of $\rho_g = 1.164 \text{ kg/m}^3$ at 30°C , the superficial gas velocity is between 0.91 and 1.01 m/s.

Results and discussion

In this investigation, numerical modelling and simulation as well as experimental studies have been carried out for validation of hydrodynamics of gas–sand multiphase flow in the fluidized bed combustor model. This validation is necessary for the optimisation of the control system in this multivariable process as well as for developing a further combustion model.

The hydrodynamic behaviours of the bubbling fluidized bed combustor were analysed by monitoring the contour plots of static pressure drop across the bed and solid volume fraction profile.

The distribution of pressure drop across the packed bed of sand particles against four inlet velocities, 0.5, 1.0, 1.25 and 1.5 m/s using the Syamlal and O'Brien drag model are shown in Fig. 3.

It can be seen that the bed height increases with the increasing of gas superficial velocity.

It has been also observed that by increasing the gas inlet velocity, the pressure drop has been increased until the condition, where the gas velocity is at minimum fluidization velocity of 1.0 m/s has been reached.

At this minimum fluidization velocity the pressure drop remains constant.

Furthermore, by further increasing of the inlet gas velocity, the void fraction increases with the bed expansion in the fluidized bed, which in turn leads to decrease in pressure drop.

The obtained volume fraction results of solid phase in Fig. 4 show the solid particles in bubbling regime at minimum fluidization velocity of 1.0 m/s.

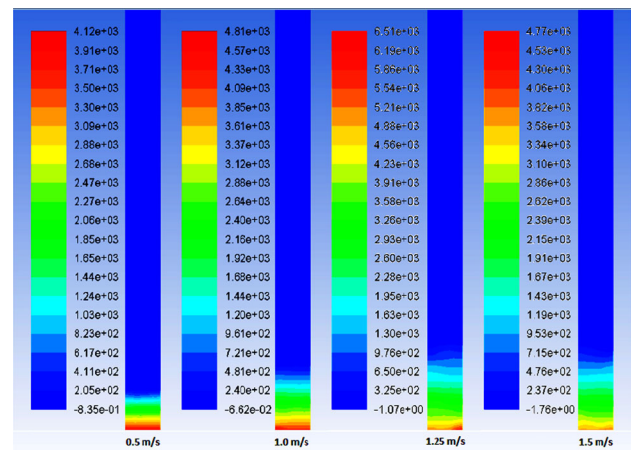


Fig. 3 Static pressure of bed materials for sand particles using the Syamlal and O'Brien drag model with four inlet velocities, 0.5, 1.0, 1.25 and 1.5 m/s

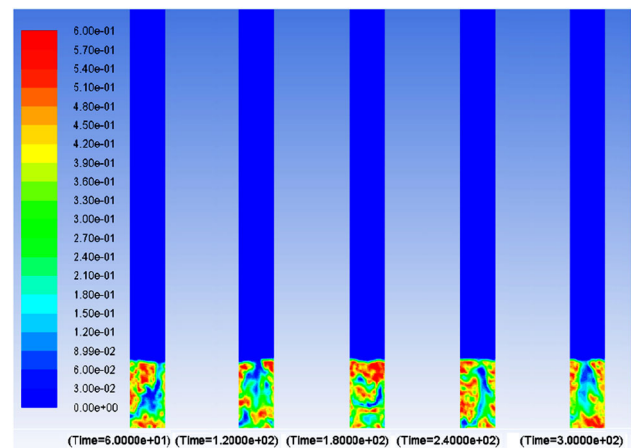


Fig. 4 Volume fraction distribution for the sand particles using the Syamlal and O'Brien drag model with four inlet velocities 1.0 m/s

At this minimum fluidization velocity, the fluidization state is stable and the bed height remains constant over the whole simulation time.

Uniform particle distribution across the distributor plate and stable gas bubbles were observed, and thus a uniform fluidization of the sand particles is achieved.

The four superficial gas velocities of 0.5, 1.0, 1.25 and 1.5 m/s used in these CFD investigations are compared to the experimentally measured minimum fluidization velocity obtained from the pilot-scale bubbling fluidized bed. Both simulations and measurements of the bubbling fluidized bed were run for 350 s. The value of minimum fluidization velocity of the pilot-scale bubbling fluidized bed combustor is approximately obtained to be at around 1.0 m/s.

In the CFD simulations, the exact value of minimum fluidization can only be obtained by performing more simulations near this value as shown in Fig. 5.



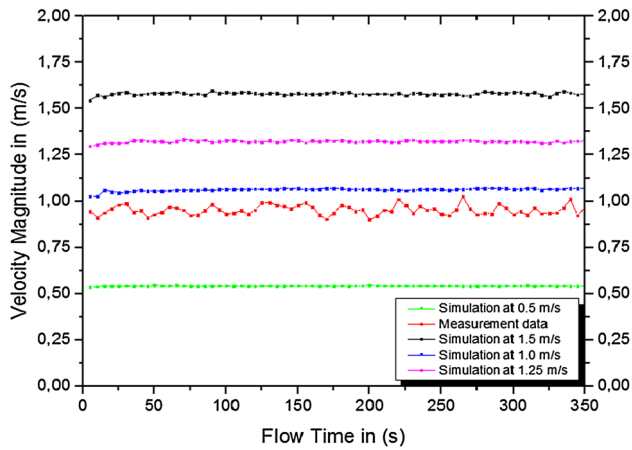


Fig. 5 Time series of superficial fluidization velocities obtained from CFD simulations compared with experimental measurements

It is shown that there is no significant difference between the minimum fluidization velocities, the simulation and measurement results are very close to each other all the time.

The pressure drop results from these simulations are plotted as a function of these gas velocities and the minimum fluidization velocity is defined as the point in which the pressure drop across the bed remains constant, and finally, the predicted pressure drop results were validated by comparing with the real data from the pilot-scale fluidized bed.

Figures 6, 7, 8 and 9 show the graphically presented pressure drop results of the bubbling fluidized bed obtained from the CFD simulations.

These predicted results have been studied by considering different superficial gas velocities, both under and above the minimum fluidization velocity. All these pressure drop results are plotted as a function of the superficial gas velocity.

It can be seen that the total static pressure drop predicted from the CFD simulations across the distributor varies between 3250 and 4350 Pa for all inlet superficial gas velocities.

The predicted pressure drop results at a superficial gas velocity of 0.5 m/s are plotted in Fig. 6.

It is observed from the Fig. 3 that the bed material height at the superficial gas velocity of 0.5 m/s remains unaffected (fixed bed), and there is no movement of sand particles. Furthermore, it can be seen from Fig. 6 that at this superficial gas velocity of 0.5 m/s, the pressure drop profile remains constant with an average of about 4000 Pa during the whole simulation period. In this fixed bed condition the gas flowing across the sand particles does not have enough velocity, which was less than the minimum

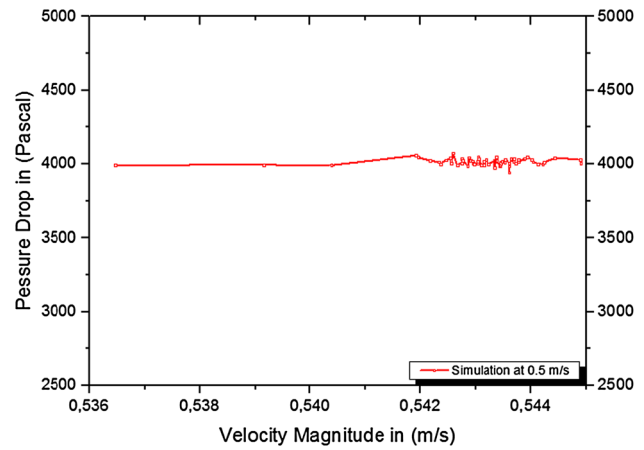


Fig. 6 Plot of pressure drop across the bed against superficial velocity of about 0.5 m/s

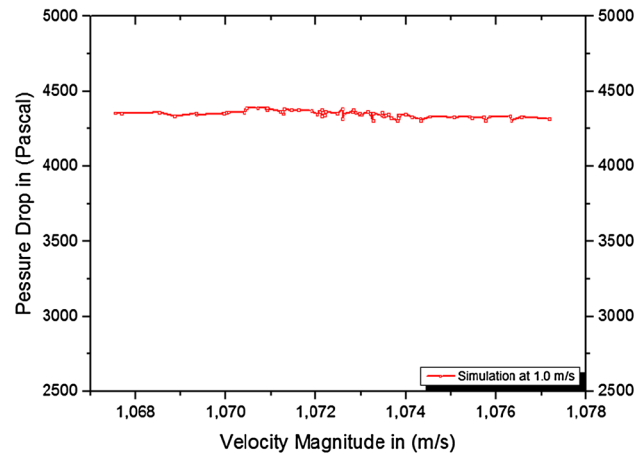


Fig. 7 Plot of pressure drop across the bed against superficial velocity of about 1.0 m/s

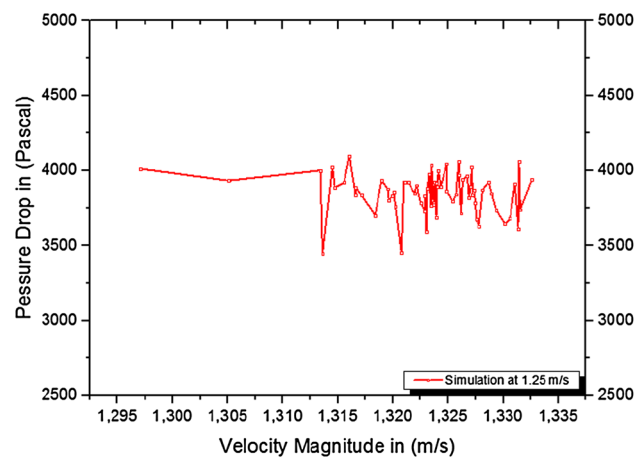


Fig. 8 Plot of pressure drop across the bed against superficial velocity of about 1.25 m/s



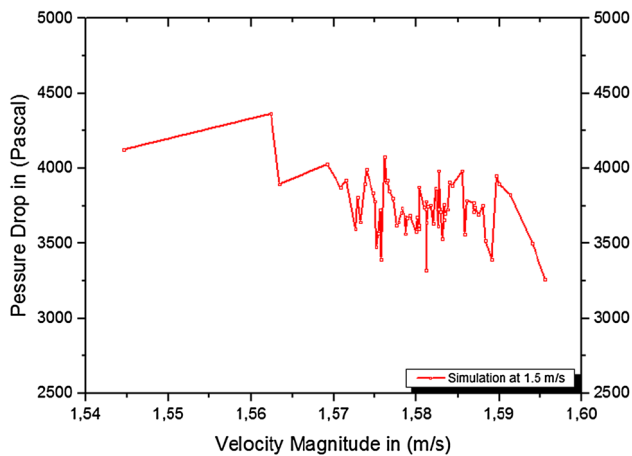


Fig. 9 Plot of pressure drop across the bed against superficial velocity of about 1.5 m/s

fluidization velocity at all obtained pressures to move the solid particles.

The pressure drop results at the superficial gas velocity of 1.0 m/s are plotted in Fig. 7.

By increasing the superficial gas velocity from 0.5 to 1.0 m/s, the pressure drop has been increased from 4000 to 4350 Pa.

It is shown that at the gas inlet velocity of 1.0 m/s, the maximum pressure drop of about 4350 Pa has been reached, and the superficial gas velocity value at which the maximum pressure drop is reached, is considered to be the minimum fluidization velocity. It can be also seen that the fluidization state is stable and the expansion ratio remains constant.

As the superficial gas velocity is increased further above the minimum fluidization velocity of 1.0 m/s to the gas velocities 1.25 and 1.5 m/s as shown in Figs. 8 and 9, the pressure drop results across the bed area has been decreased. This decrease in pressure drop after reaching the steady state fluidization is explained by the increase of void fraction. Furthermore, a transformation from bubbling fluidization to slugging regime has been observed for both simulations at these superficial gas velocities, which are higher than the minimum fluidization velocity of 1.0 m/s.

The simulation results of static pressure drop across the distributor at the superficial gas velocity of 1.25 m/s are plotted in Fig. 8.

It can be observed that an increase in the gas flow rate above the minimum fluidization velocity will directly result in a decrease of the pressure drop of bed. A sudden decrease in the pressure drop is observed. Furthermore, an unstable regime of fluidization resulting in large pressure fluctuations has been also observed.

When the velocity of a gas through a bubbling fluidized bed is increased above the minimum bubbling velocity, the

bubble size increases and frequently split and coalesce passing through the bed as slug.

The passage of these gas slugs produce large pressure fluctuations inside the fluidized bed. The fluctuations in pressure drop were caused by the slugging flow regime.

The pressure drop fluctuation results at a superficial gas velocity of 1.5 m/s are plotted in Fig. 9.

It is observed that with a further increase in gas velocity from 1.25 to 1.5 m/s, the predicted pressure drop has been further decreased. Larger and higher amplitude of pressure drop fluctuations has been observed, which results in further instabilities of the fluidization. These predicted instabilities in the fluidization of the pilot-scale bubbling fluidized bed are caused by the slugging-flow regime.

Figure 10 shows a comparison of the CFD numerical simulation results plotted in Figs. 6, 7, 8 and 9 with the measurements. All obtained simulation results were run for 350 s and validated against experimental results.

It can be seen that at the superficial gas velocity of 1.0 m/s, both experimental measurements and CFD simulation give approximately the same pressure drop values between 4250 and 4350 Pa. In contrast, all other simulation results at under and below the minimum fluidization velocity have predicted a lower pressure drop values than measurements. At minimum fluidization velocity, all obtained results have shown to promote a good prediction of pressure drop across the bed during whole operation time. Therefore, the pressure drop predicted by the CFD simulation at a superficial velocity of 1.0 m/s using Syamlal and O'Brien drag model agreed reasonably well with the experimental measurements. Furthermore, there is no significant difference between the experimental minimum fluidization velocity of the test rig and the minimum superficial gas velocity obtained based on the predicted pressure drop results. Finally, it can clearly be seen that Syamlal and O'Brien drag function gives a good prediction

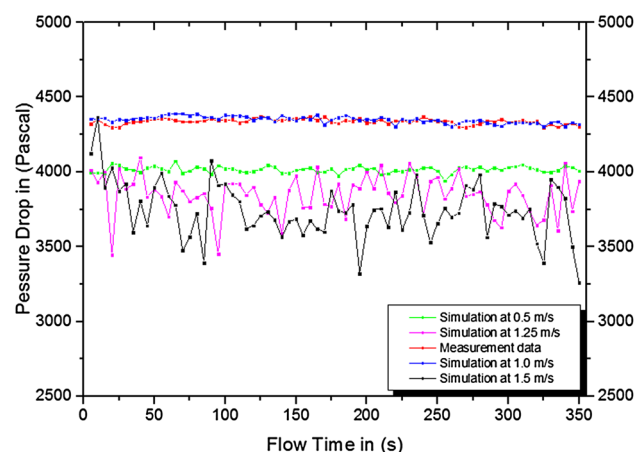


Fig. 10 Comparison of experimental measurements and CFD simulated results for pressure drop at different velocities



in terms of pressure drop and also, Syamlal and O'Brien drag law correctly predicts the minimum fluidization conditions.

Conclusion

The CFD model and simulation are created as a realistic representation of the actual pilot-scale bubbling fluidized bed. The validation of the predicted results has been based on experimental measurements. Therefore, the predicted pressure drop results were compared to the experimental measurements obtained from the pilot-scale bubbling fluidized bed combustor. The value of the minimum fluidization velocity U_{mf} , at which the pressure drop across the bed reaches a maximum value and the gas–solid flow achieves a uniform and stable fluidization regime is found to be 1.0 m/s. This minimum fluidization gas velocity predicted from the numerical simulation is approximately equal to the superficial gas velocity of the bubbling fluidized bed combustor. At this minimum superficial gas velocity, the pressure drop results predicted from the simulation using the Syamlal and O'Brien drag model were similar to the pressure drop measurements. Therefore, the predicted pressure drop results from the three-dimensional CFD simulation including the minimum fluidization velocity were found to agree well with the experimental pressure drop data across the bed. These findings show that the CFD simulation using the Syamlal and O'Brien drag model is capable to predict the hydrodynamics in fluidized bed combustors, and thus the proposed model provides a useful basis for further works on the development of the CFD simulation for the combustion part as well as for future control strategies of the process.

The next study will investigate the influence of different solid particle diameters on pressure drop in a large-scale industrial sewage sludge fired bubbling fluidized bed.

Compliance with ethical standards

Conflict of interest The authors declare that there are no conflicts of interest regarding the publication of this article.

Open Access This article is distributed under the terms of the Creative Commons Attribution 4.0 International License (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license, and indicate if changes were made.

References

- Lundberg, J.: CFD study of a bubbling fluidized bed. Master thesis, Telemark University College Norway (2008)
- Karmakar, M.K., Haldar, S., Chatterjee, P.K.: Studies on fluidization behaviour of sand and biomass mixtures. *Int. J. Emerg. Technol. Adv. Eng.* **3**(3), 180–185 (2013)
- Schreiber, M., Asegehegn, T.W., Krautz, H.J.: Numerical and experimental investigation of bubbling gas–solid fluidized beds with dense immersed tube bundles. *Ind. Eng. Chem. Res.* **50**, 7653–7666 (2008)
- Armstrong, L.-M.: CFD modelling of the gas–solid flow dynamics and thermal conversion processes in fluidised beds. Ph.D. thesis, University of Southampton (2011)
- Sahoo, P., Sahoo, A.: A comparative study on fluidization characteristics of coarse and fine particles in a gas–solid fluidized bed. *CFD Anal. Int. J. Eng. Sci. Innov. Technol.* **3**, 246–252 (2014)
- Vejahati, F., Mahinpey, N., Ellis, N., Nikoo, M.B.: CFD simulation of gas-solid bubbling fluidized bed. A new method for adjusting drag law. *Can. J. Chem. Eng.* **87**, 19–30 (2009)
- Herzog, N., Schreiber, M., Egbers, C., Krautz, J.H.: A comparative study of different CFD-codes for numerical simulation of gas–solid fluidized bed hydrodynamics. *Comput. Chem. Eng.* **39**, 41–46 (2012)
- Taghipour, F., Ellis, N., Wong, C.: Experimental and computational study of gas–solid fluidized bed hydrodynamics. *Chem. Eng. Sci.* **6**, 6857–6867 (2005)
- Hamzehei, M.: CFD modelling and simulation of hydrodynamics in a fluidized bed dryer with experimental validation. *ISRN Mech. Eng.* (2011). doi:[10.5402/2011/131087](https://doi.org/10.5402/2011/131087)
- Ramesh, P.L.N., Raajenthiren, M.: A review of some existing drag models describing the interaction between the solid–gaseous phases in a CFB. *Int. J. Eng. Sci. Technol.* **2**(5), 1047–1051 (2010)
- Almutterah, A.M.: CFD modeling of the hydrodynamics of circulating fluidized bed riser. Master thesis, The University of British Columbia (2006)
- Esmaili, E., Mahinpey, N.: Adjustment of drag coefficient correlations in three dimensional CFD simulation of gas–solid bubbling fluidized bed. *Adv. Eng. Softw.* **42**(6), 375–386 (2011)
- Asegehegn, T.W., Schreiber, M., Krautz, H.J.: Numerical simulation of dense gas–solid multiphase flows using Eulerian–Eulerian two-fluid model. In: Zhu, J. (ed.) *Computational simulations and applications*. InTech, Rijeka (2011)
- Azadi, M.: Multi-fluid Eulerian modelling of limestone particles elutriation from a binary mixture in a gas–solid fluidized bed. *J. Ind. Eng. Chem.* **17**, 229–236 (2011)
- Fan, R.: Computational fluid dynamics simulation of fluidized bed polymerization reactors. Ph.D. thesis, Iowa State University (2006)
- England, J.A.: Numerical modelling and prediction of bubbling fluidized beds. Master thesis, Virginia Poly Technique Institute and State University (2011)
- Abirha, T.: Design and development of fast pyrolysis fluidized bed reactor for bio-oil production. Master thesis, Addis Ababa University (2011)
- Rangelova, J.: Auftriebsverhalten von Feststoffpartikeln in Wirbelschichten. Ph.D. thesis, Otto von Guericke University of Magdeburg (2002)
- Sobrinho Fernandez, C.: Experimental study of a bubbling fluidized bed with a rotating distributor. Ph.D. thesis, Carlos III University of Madrid (2012)
- Passos, M.L., Barrozo, M.A., Mujumdar, A.S.: *Fluidization Engineering Practice*. Laval, Canada (2013)
- Teaters, L.: A computational study of the hydrodynamics of gas–solid fluidized beds. Master thesis, Virginia Poly Technique Institute and State University (2012)

